U.P.B. Sci. Bull., Series D, Vol. 71, Iss. 4, 2009 ISSN 1454-2358

OPTIMIZATION APPROACH TO CONCEPTION OF A MECHANICAL PART USING CAD/FEM TECHNIQUES

Ionuț Gabriel GHIONEA[1]

Lucrarea prezintă o metodologie practică de analiză cu elemente finite pentru optimizarea concepției şi proiectării pieselor mecanice. Procedura de lucru foloseşte mediul CAD-FEM CATIA, un model 3D, o forță aplicată acestuia, constrângeri şi unele tehnici de optimizare. Calculele necesare pentru piesa analizată sunt realizate prin metoda elementelor finite şi interpretate cu ajutorul unui program scris în Visual Basic. Scopul abordării este de a determina tensiunile rezultante după aplicarea forței şi de a crea o regulă automată pentru limitarea valorii acesteia. În lucrare sunt prezentate şi unele aspecte teoretice FEM.

The paper presents a practical application methodology in FE analysis for the optimization of the mechanical parts' conception and design. The working procedure uses the CATIA CAD-FEM environment, a 3D model, a loading force, restraints and some optimization techniques. The necessary calculations for the analyzed part are made using the finite element method and interpreted with the aid of a Visual Basic program. The purpose of this approach is to determine the resulting stresses under a loading force and to create an automatic rule for the force limitation. In the paper are presented theoretical aspects in the FE analysis.

Keywords: CAD model, finite element method and analysis, Von Mises stresses, loading force limitation, displacements, reaction and sensor creation

1. Introduction

The finite element method (FEM) is one of the most used methods available for different simulations and calculations in the engineering field. This method and the programs based on it become fundamental components in the modern computer aided design systems, for all engineering activities where high performance is required. The main purpose of this paper is to present a practical application using computer aided design and the finite element analysis to elaborate a correct and efficient calculus model to improve the mechanical components conception [3]. The topics in the paper vary from theoretical aspects of CAD and FEM to a practical problem of modeling, analysis and results interpretation, with many step by step explanations, helping the reader to understand the problems and to draw clear and convincing conclusions [1], [2].

[1] Lecturer, Dept. of Machine Manufacturing Technology, University POLITEHNICA of Bucharest, Romania, e-mail: ionut76@hotmail.com

2. CAD/FEM – basic concepts and advantages in use

One of the major advantages in the finite element method is the simplicity of its basic concepts. It is very important that the FEM user correctly understand the concepts, because they include hypotheses, simplifications, generalizations. Using a CAD-CAE system it is possible to replace the real external forces by efforts (forces, accelerations, torsion moments, masses etc.) to which they are statically equivalent, but this equivalence is not allowed in the theory of elasticity.

To perform a finite element analysis, the user must develop a calculus model of the analyzed structure. These models are only approximate mathematical models of the structure. There are no algorithms and general methods for developing a unique model that approximates the real structure [1].

The model consists of lines, planes or curved surfaces and volumes, created in a 3D CAD environment. In this stage of development, the model is continuous, with an infinite number of points, as the real structure. The main goal of FEM is to obtain the finite element mesh, transforming the continuous structure into a discrete model, with a finite number of points [2], [3]. This operation is done using a mesh for the model, correct from an engineering point of view, knowing the stresses and displacements in a certain number of points inside the structure is generally enough to characterize the mechanical behaviour of the whole structure.

The finite element method defines these data only in the nodes of the model and calculates their values in these points. That is why the meshing process must be performed in such a way as to have a number of nodes large enough in the areas of great interest in order to achieve a satisfactory approximation for the geometry of the structure and for the boundary and loading conditions [1], [5].

The points defined in the mesh are called nodes. The primary unknowns of FEM are the parameters defined in these nodes, and their values are the analysis results. The data identified in the nodes can be displacements (displacement model) or stresses (stress model). The meshing process divides the model into a certain number of quadrilateral or triangular fragments, called finite elements. These elements are assembled together in common nodes, also called vertices. The FEM study the finite element as a single piece in interaction with the other elements only in nodes. Thus, the study of the real structure is replaced with the study of the ensemble of finite elements obtained by meshing [2].

In the finite element method practice, the role of the material characteristics is very important. Each finite element is an ensemble of conditions and hypotheses and should be used with care and only after a complete study of the environment where the real structure is functioning: loadings, stress type, interaction with other elements etc.

The techniques used in a CAD/FEM approach for the optimization of a mechanical structure (part, assembly) conception have many advantages [1], like:

- general use: FEM is a numerical method used for solving problems in mechanics of deformable bodies, heat transfer, biomechanics etc. The loading case can be static, dynamic, nonlinear, time dependent;

- litheness: to solve a problem with this method, the user is free to elaborate the calculus model to meet his requirements. After this elaboration, often it is very simple to modify it, the goal is to obtain an enhanced variant of the initial model;

- simplicity of the basic concepts: the method doesn't require the user to have a good knowledge of mathematics, but just an engineering background. The basic concepts are easy to understand for a correct use;

- existence of the computer programs: the use of the finite element method requires a large amount of numerical calculations;

For these reasons, many companies implemented in their engineering systems procedures for CAD modelisation and FE calculations. Also, there are some FE specialized programs, offering the user a good percent of stability and safety. The use of the computer programs reduce the work time, enable automatic meshing and verification of the model. The results obtained from the CAD/FEM can be presented as images, tables, listings, drawings, diagrams, animations etc.

3. The practical application

This paper presents a FE analysis of a support type part, from a mechanical assembly, having the draft design shown in Fig. 1. To fulfill the functional role, the model contains assembly surfaces with other components, but also with a connection surface, on which the user will apply a loading force.

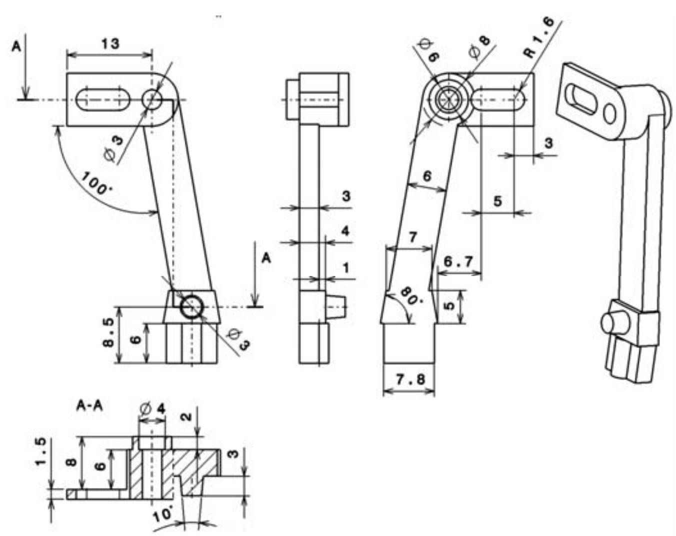

Fig. 1. The 3D CAD model of the support type part

After solid modeling using the module CATIA Part Design, the part is considered as manufactured of steel, with the following elastic constants and mechanical characteristics (Fig. 2): Young's modulus (2×10^{11} N/m^2), Poisson' ratio (0.266), density (7860 kg/m3), coefficient of thermal expansion (1.17×10^{-5} K), yield strength (2.5×10^{8} N/m^2) [2].

PartBody\Material	Steel
\`Steel\Steel.1.1\Young Modulus\`	2e+011N_m2
\`Steel\Steel.1.1\Poisson Ratio\`	0.266
\`Steel\Steel.1.1\Density\`	7860kg_m3
\`Steel\Steel.1.1\Thermal Expansion\`	1.17e-005_Kdeg
\`Steel\Steel.1.1\Yield Strength\`	2.5e+008N_m2

Fig. 2. Material properties for the part

To start the FE analysis, the user can access the CATIA Generative Structural Analysis module to set up the type of statical analysis as Static Case.

Although the program defines the default network of nodes and elements, it may be edited, allowing the user to establish the size of the finite element, maximum tolerance between the meshed model and the model used in real analysis and the element type [1], [5].

On each surface of the bearings from the bottom of the part, the user applies the Clamp restrains as shown in Fig. 3.

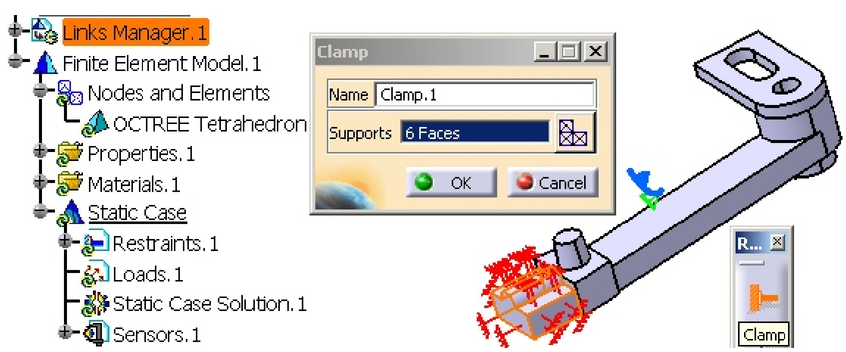

Fig. 3. Restraints application on the bottom bearing surfaces

On the surface of the bore from the upper zone of the part structure, it is applied a distributed force of 150 N from interior to exterior in the opposite direction of the X axis.

The force is represented by a four red arrows symbol, and characterized by a value and direction. All the parameters can be introduced in the appropriate fields from the dialog box as Figure 4 shows. After the restraints and the loading force are established, the next step is to calculate the model behaviour running the Compute routine.

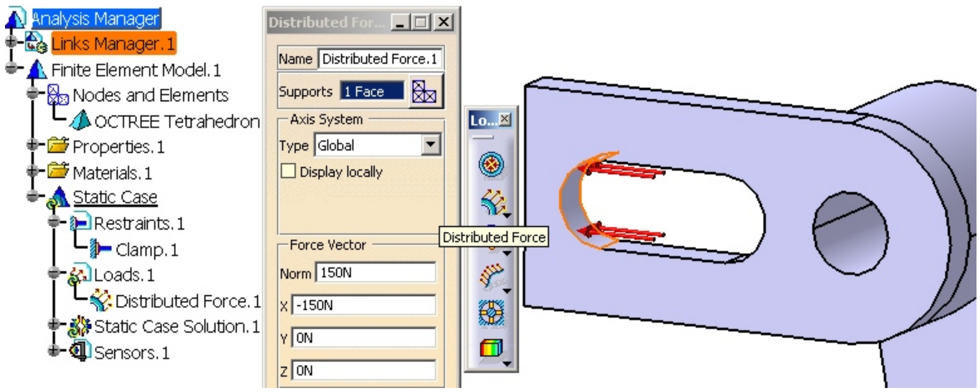

Fig. 4. Application of the loading force

At the end of the calculus, the user may decide to view the results in a specific representation. Thus, Fig. 5 shows four output results as images. A variety of parameters might be studied, but the present approach analyses the structure behaviour considering the displacement, principal stress and precision [2].

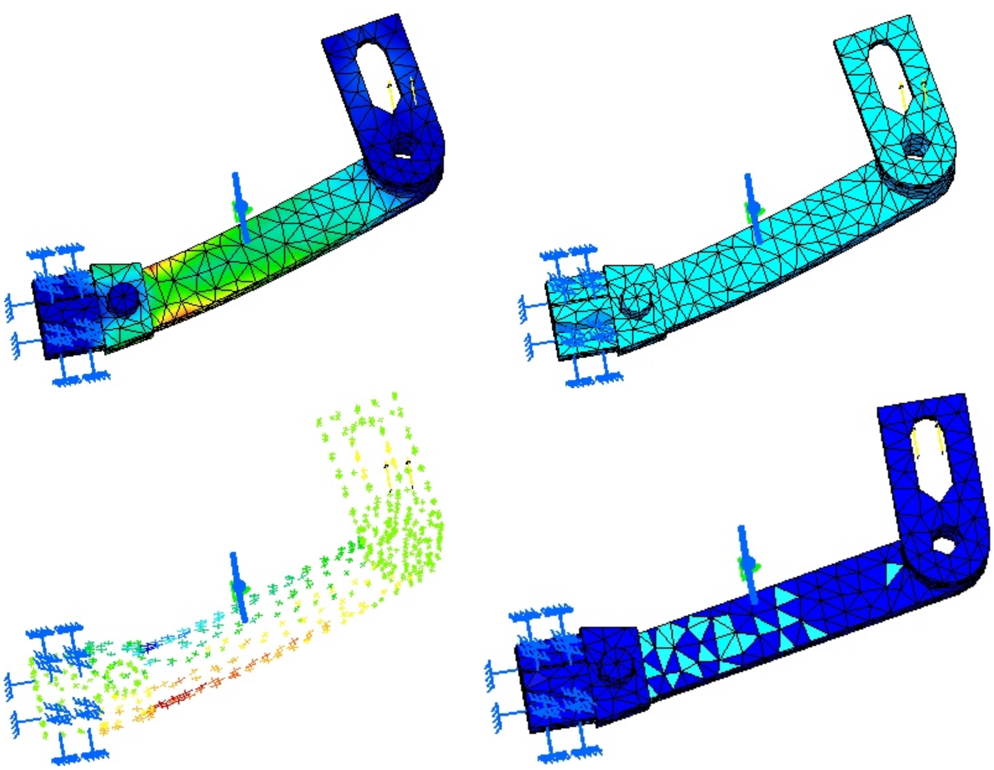

Fig. 5. Image results of the FEM analysis

In order to identify the maximum and minimum stress values the Information instrument is used to show the results. Fig. 6 presents the window with stress value, and the colour scale of Von Mises image. The lowest stress values correspond to the bottom of the scale, close to blue; the higher are the red ones at the top of the scale; between these are quite safe yellow and green shades.

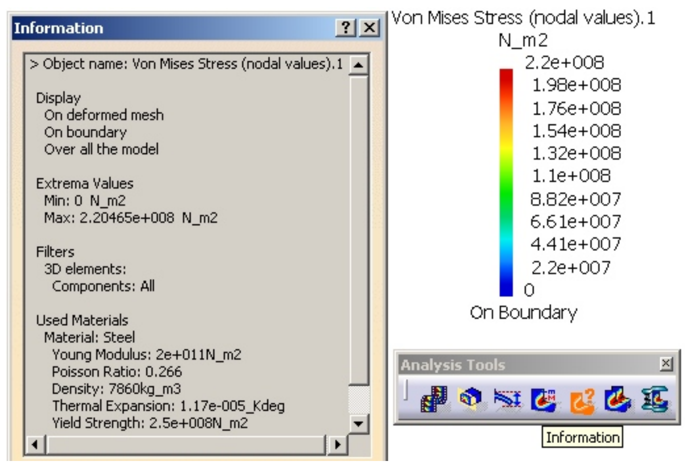

Fig. 6. Informations for the Von Mises stresses distribution over the part

The user can understand how the stresses are distributed on the part also from the way the colors are displayed on the FEM model [3], [5]. Knowing the yield strength of the material (2.5×10^8 N/m^2), it can be considered that the part structure will resist to a distributed loading force of 150 N, but the operational safety is nearby the maximum limit and an increase of more than 10% is dangerous and not recommended. To verify the accuracy of the results a specific test with the Precision instrument is performed in order to locate the zones containing the biggest errors. The calculated global error is 41.9%; such error seems to be high but it indicates, in fact, all the differences between the proposed virtual model and the real structure.

Therefore, it is important to optimize the model running an adaptivity instrument and filling in a suitable value – for the start: 25%. To reiterate the computation in 5 steps it is imperative for the analysis to refine the model and to reach the established goal. The mesh network is also refined. Definitely, all these improvements of the virtual model increase the computing time [2], [6]. Analysis of the optimized model results shows a lower error percent, about + 4.58% more than the initial target of 25%.

Overall, an improvement of + 12.36% has been achieved, but regarding the Von Mises outputs, the new model analysis shows an increased maximum stress value, from 2.20×10^8 N/m^2 to 2.34×10^8 N/m^2.

For a higher precision and improved model, the user may continue an iterative analysis and to refine the model. The goal is to decrease the global error, but the outputs of the maximum stress must not exceed the admissible yield strength of the considered steel. In such cases, the user should be informed of the overruns and the program must provide a solution to change the part's material, knowing that the geometric shape of the part is imposed and cannot be changed. The values for stresses, displacements, estimated global and local errors, calculated during the application represent important parameters that may be involved in formulae, rules and reactions, but first they must be identified using the sensors. With their aid, the user finds out different concise information on the results of the analysis [2]. In this application, he is interested in the value of maximum stress (Von Mises Stress) and maximum displacement of the part, as a result of loading it with a set consisting of a restriction and a distributed force.

From the values showed by the displacement sensor results that some surfaces of the part are displaced over a max. distance of 0.25 mm (Fig. 7).

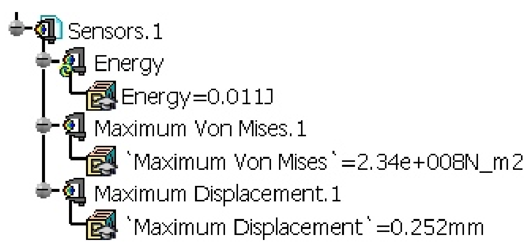

Fig. 7. Different sensors created by user after the FEM analysis

This displacement is accepted by the user, but it must not exceed a required value of 0.35 mm, set by the functional role of the part in the assembly. From the calculations previously done, the part resist to a loading force of 150 N, with a material having the yield strength equal to 2.5×10^8 N/m^2. Increasing the amount of force will produce a maximum stress, greater than this value, and the part's displacements will enter in the plastic domain, not in the elastic one. This situation is not accepted due to the functional role of the part in the assembly.

Knowing the occured displacement restrictions and stresses, the user can write a reaction using the module CATIA Knowledge Advisor. This reaction will monitor the changing values of the applied force and will impose some parameters modifications, as appropriate. The reaction activation condition is set upon the modification of the loading force.

The reaction code is written in the Visual Basic language using various parameters and syntax particularities of the CATIA Knowledge Advisor module. The program is inserted line by line in a complex code sequence presented in Fig.8 and explained as follows.

```
if `Finite Element Model.1\Distributed Force.1\Force Vector.1\Force.1` <-150N
{Message("The force value on the negative direction of the X axis|has become: ",
abs(`Finite Element Model.1\Distributed Force.1\Force Vector.1\Force.1`),"N", " The FEM
calculations will be resumed.")
if `Finite Element Model.1\Maximum Von Mises.1\Maximum Von Mises` > 2.5e+008N_m2
{Message("The maximum tension value has become: #  |and it exceeds the yield strength
value of 2.5e+008N_m2", `Finite Element Model.1\Maximum Von Mises.1\Maximum Von
Mises`)
`Steel\Steel.1.1\Yield Strength` =3.5e+008N_m2
Message("A new steel is chosen, having the yield strength of 3.5e+008N_m2.")
if `Finite Element Model.1\Maximum Displacement.1\Maximum Displacement` > 0.35mm
{Message("The maximum deformation exceeds 0.35mm.|This deformation is:",`Finite
Element Model.1\Maximum Displacement.1\Maximum Displacement`)
`Finite Element Model.1\Distributed Force.1\Force Vector.1\Force.1`=-205N
Message("The force was limited to the value of: 205N") }}}
else if `Finite Element Model.1\Distributed Force.1\Force Vector.1\Force.1` >= -150N
{`Steel\Steel.1.1\Yield Strength` =2.5e+008N_m2}
```

Fig. 8. Visual Basic code sequence for the force limitation and part's material change

First, the applied force value is compared with 150 N, initially established, for this value the results were previously presented. The minus sign means the negative direction of the X axis. Then, the force value is changed, if the 150 N value is exceeded, an information message is showed (Fig. 9) and the FEM calculations are resumed.

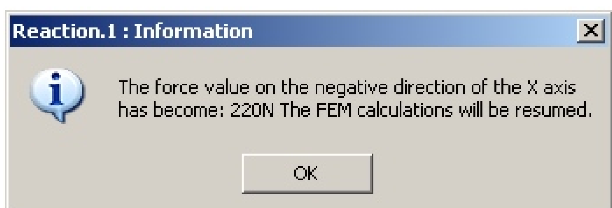

Fig. 9. The user is informed that the loading force has been changed

After this step, the maximum Von Mises stress value is compared with the yield strength of the current steel (2.5×10^{8} N/m^{2}). If the comparison is true, a new information message is showed (Fig. 10), containing the calculated stress and the user finds out that the yield strength was exceeded.

Fig. 10. The program calculates the maximum stress and compares it with the yield strength

As a solution, the reaction proposes a new steel, having the yield strength equal to 3.5×10^{8} N/m^2 and informs the user about that (Fig. 11).

Fig. 11. A new steel material is automatically chosen

For the new conditions, the program calculates the maximum displacement value and compares it to the imposed value of 0.35 mm. If the condition is acomplished, an information message is displayed (Fig. 12).

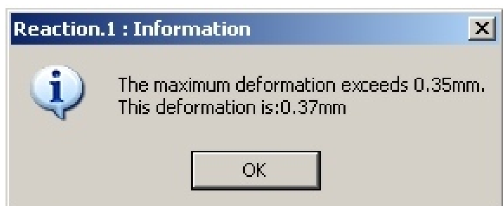

Fig. 12. The new displacement is calculated

A greater value than 0.35 mm is not accepted (condition initially imposed by the assembly which contains the analyzed part), the solution being the automatic reduction of the force value which becomes equal to 205 N. This value is, also, the maximum suported by the part in it's assembly, in the conditions of a new steel material.

Fig. 13. The loading force is limited to the maximum value

Next, an information message is displayed (Fig. 13) with the automatic force limitation, but, if the force value returns to a value equal or smaller than 150N, the first steel material (2.5×10^{8} N/m^2) is kept. This condition is imposed by the steel price, lower in the first case. All the sensors are now updated (Fig. 14) by the user using the contextual menu of each one.

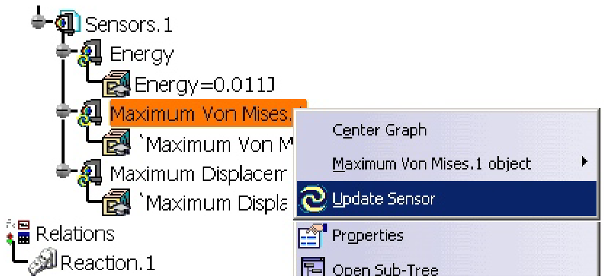

Fig. 14. Updating the sensors

Also, the results (Von Mises Stress, Estimated local error, Translational displacement vector etc.) are updated.

4. Conclusions

As a conclusion, in this approach, if, for example, a force value of 220 N is used, the reaction detects a maximum stress greater than the yield strength of 2.5×10^8 N/m^2 (of the initial chosen steel) and is changing this steel with another, having the yield strength equal to 3.5×10^8 N/m^2. The methodology presented in this paper realise an optimization of the mechanical part conception and design by an automation of the analysis process, exempting the user of taking care of the limiting the applied loading force or to change the material in certain situations, enabling him at the same time to focus on the stages of design, establishing restrictions, loads and, especially, in the final stage, on the interpretation of results.

R E F E R E N C E S

[1]. *N. I. Constantinescu, Şt. Sorohan, Şt. Pastramă,* The practice of finite element modeling and analysis. Editura PRINTECH, Bucureşti, ISBN 978-973-718-511-2, 2006.
[2]. *I. Ghionea,* Some considerations about the methodology and results for the finite element analysis of a mechanical assembly, Proceedings of the 16-th International Conference on Manufacturing Systems, Editura Academiei Române, ISBN 1842-3183, 2007, pp. 77-80.
[3]. *C. Ispas, I. Ghionea,* Stude of computer design and management in the conception and development phases of a product. Optimum Technologies, Technologic Systems and Materials in the Machines Building Field, University of Bacău, ISSN 1224-7499, 2001.
[4]. *P. Moreira et. al.,* Fatigue life prediction of a cracked lap splice specimen using fracture mechanics parameters. U.P.B. Sci. Bulletin, Series D, Vol. 71, Iss. 3, ISSN 1454-2358, 2009.
[5]. *C. Zienkiewicz, L. Taylor,* The finite element method. 5th edition. Vol. 1. The Basis Butter-worth – Heinemann, Oxford Auckland Boston, ISBN 07506549, Published CIMME, 2000.
[6]. *** CATIA v5 – User Guide, 2008.